COMPUTATIONAL FLUID DYNAMICS OF AEROFOIL SECTIONS

A thesis submitted

By

DEEPAK PANDA
(Roll No: 111ME0326)

For the necessity of the degree

of

BACHELOR OF TECHNOLOGY
in
MECHANICAL ENGINEERING

Under the guidance of

PROF. J. SRINIVAS

DEPARTMENT OF MECHANICAL TECHNOLOGY

NATIONAL INSTITUTE OF TECHNOLOGY ROURKELA

ODISHA, INDIA

NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA

CERTIFICATE

This is to confirm that the Project Report entitled, "Computational Fluid Dynamics of Aerofoil Sections" put together by Deepak Panda for the prerequisites for the honour of the Degree of Bachelor of Technology in Mechanical Engineering at National Institute of Technology, Rourkela (Deemed college) is a bona fide work attempted by him under my watch and direction. The subject included in the proposal have not been submitted to whatever other Institute for the honour of any degree which is best as far as anyone is concerned.

Prof. J Srinivas
Department Of Mechanical Engineering,
National Institute of Technology,
Rourkela-769008, Orissa (India

Place: NIT Rourkela
Date:

ACKNOWLEDGEMENTS

I would want to demonstrate my most profound appreciation to my respectable aide Prof. J. Srinivas for his bolster, direction, care, inspiration and tolerance all through the examination period. I admire his ability and the consolation which he offered to me as the year progressed. All credit goes to him who made my task effective. Without his assistance and commitment, I would not have possessed the capacity to finish my venture work and accomplish craved results and complete my four year certification.

Aside from him I might likewise want to thank each resources of my mechanical designing office for their assistance and inspiration and care all through the whole venture period.

I might likewise want to thank all the individuals who are straightforwardly or in a roundabout way included in my undertaking work and aided in effectively finishing this task work.

Deepak Panda

Roll No: 111ME0326

Mechanical Engineering Department

CONTENTS

ABSTRACT

Airfoil plays an important role in any aircraft and chooses whether the lift power is adequate to adjust the heaviness of the body or not and the amount of drag power is being connected on the body. The main objective of this investigation is to analyze the flow behaviour around the airfoil body and to calculate the performance coefficients at higher Reynolds Number (1.04×10^5) and angle of attack varying from -5 degrees to 20 degrees. The section-lift and drag coefficient for airfoils are obtained by analyzing the measured pressure distribution on the airfoil surface. The simulations were computed in the software ANSYS 15.0 FLUENT. FLUENT software screens the surface weight and gives a visual presentation of the element changes connected with shifting approach. By plotting the bend L/D, yields the most proficient aerofoil approach. Lift increments as the approach ascends from -5 to 20 degrees and at a specific point, most extreme lift is produced. On the off chance that the approach is expanded any further, drag turns into the overwhelming component and the area enters the stall mode. A simple approach called PANEL method is used to validate the results obtained from FLUENT software. This approach can be further used as function in inverse design of aerofoil sections for desired pressure distributions.

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

l= span

c= chord length (m)

α= angle of attack

D= Drag Force

L= Lift Force

M= Moment

U_α= Free Stream Velocity

ρ= Air density

μ= Dynamic viscosity (Ns/m^2)

p_α= Free stream pressure (Pa)

p= local pressure (Pa)

Re= Reynolds no ($\rho U_\alpha c/\mu$)

C_p= pressure coefficient ($(p-p_\alpha)/ (0.5\rho U_\alpha^2)$)

C_l= lift coefficient ($L/(0.5\rho U_\alpha^2 C_l)$)

C_d= drag coefficient ($D/(0.5\rho U_\alpha^2 C_d)$)

CHAPTER 1

INTRODUCTION

In the earliest days, when man was yet living in the lap of nature, the main method for velocity was his legs. Continuously, we have accomplished quicker and richer methods for voyaging, most recent being the air transport. Since, its innovation planes have been getting more fame as it is the quickest method of transportation accessible. It has additionally picked up fame as a war machine since World War II. This prominence of air transport has prompted numerous new innovations and exploration to grow quicker and more conservative planes. This task is such an endeavour to decide how we can infer most extreme execution out of an aerofoil segment.

An aerofoil is a cross-area of wing of the plane. Its fundamental occupation is to give lift to a plane amid departure keeping in mind in flight. Yet, it has likewise a reaction called drag which restricts the movement of the plane. The measure of lift required by a plane relies on upon the reason for which it is to be utilized. Heavier planes oblige more lift while lighter planes oblige less lift than the heavier ones. Accordingly, contingent on the utilization of plane, aerofoil area is resolved. Lift compel additionally decides the vertical increasing speed of the plane, which in turns relies on upon the flat speed of the plane. Hence, deciding the coefficient of lift one can ascertain the lift compel and knowing the lift drive and obliged vertical increasing speed one can focus the obliged level speed.

Aircraft wings which are horizontal and vertical stabilizers, helicopter rotor blades, propellers, fans, compressors turbines all have aerofoil designs. Even sails, swimming and flying creatures employ aerofoils. An airfoil-shaped wing can create down force on an automobile or other motor vehicle, improving traction.

A laminar flow wing has a greatest thickness in the centre camber line. It demonstrates a negative weight inclination along the flow has the same impact as decreasing the rate when we dissect the Navier–Stokes mathematical statements in the straight administration. So on the off chance that we keep up greatest camber in the centre, a laminar flow over a bigger rate of the wing at a higher velocity can be attained to. Nonetheless, with particles on the wing, this does not work. Since such a wing slows down more effectively than others.

1.1 BASIC TERMINOLOGY

Fig.1.1 shows the aerofoil cross-section with a flow stream passing at a particular angle of attack.

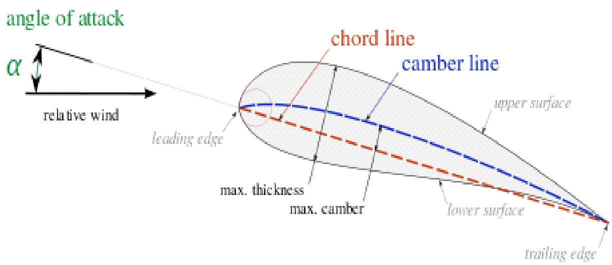

Fig. 1.1 Airfoil Section

The various terms in airfoils are described below:

Pressure surface is the lower surface with comparative higher static pressure than the suction surface.

Suction surface is the upper surface with higher velocity and lower static pressure.

The geometric terms related to airfoil is described below:

The *trailing edge*- It is the point of maximum curvature at the rear of the airfoil.

The *leading edge*- It is the point located at the front of an airfoil that has maximum curvature.

Chord. It is the length of the chordline. Features of the airfoil are described in percentage of the chord measured from leading edge to trailing edge.

The *chord line*- It is the straight line connecting leading and trailing edges.

The *chord length* is the length of the chord line.

The parameters defining shape of an aerofoil are:

The *thickness* of an airfoil varies along the chord. It may be measured two ways: one which is measured perpendicular to the camber line and other which is measured perpendicular to the chord line.

Mean camber line (MCL). The line which is exactly in the middle of upper and lower surfaces is mean camber line. The MCL for a cambered airfoil generally is above the chordline whereas it is coincident with the chordline in case of symmetric aerofoils.

Maximum Thickness- It is the maximum separation from the bottom edge to the top edge. It is generally 0.12c or 12% of the chord

Maximum camber. - The maximum distance of the MCL from the chordline. Maximum camber is generally expressed as a % or fraction of the chord.

Leading Edge Radius - The radius of a circle that produces the leading edge curvature.

Stalling speed is the slowest speed at which aircraft can fly in straight and level flight. It is defined in terms of the maximum lift coefficient as follows :-

$$L=\frac{1}{2}\rho v^2 SC_l \quad \text{or} \quad v=\sqrt{\frac{2w}{\rho SCl}} \quad \text{where } L=W \tag{1.1}$$

Therefore, higher the lift, lesser the stalling speed.

Angle of attack is defined as the angle between the chordline and the relative wind or flight path.

Total aerodynamic force (TAF) is the total force on the airfoil produced by the airfoil shape and relative wind.

Lift is the perpendicular component of TAF to the relative wind or flight path.

Drag is the parallel component of TAF to the relative wind or flight path.

Flap is an artificial high lift providing device attached to the aerofoil section at trailing edge. When flap is deflected downwards, the lift coefficient increases due to increase in camber of aerofoil sections.

1.2 AIRFOIL PROFILES DESIGNATION

The NACA aerofoils are aerofoil shapes for air ship wings which are created by the National Advisory Committee for Aeronautics (NACA). The state of the NACA aerofoils is depicted regarding arrangement of digits taking after "NACA". The numerical code can be gone into mathematical statements of aerofoil to produce the cross-area of the aerofoil and compute its properties.

The NACA aerofoil arrangement, the 4-digit, 5-digit, and adjusted 4-/5-digit, were created utilizing scientific comparisons that portray the camber of the mean- line (geometric centreline) of the aerofoil area and additionally the segment's thickness conveyance along the length. Later, including the 6-Series, confused shapes were inferred utilizing hypothetical techniques. Prior to the National Advisory Committee for Aeronautics (NACA) added to these arrangements, aerofoil configuration was somewhat discretionary outlines aside from past involvement with known shapes and experimentation with adjustments to those shapes. Distinctive NACA aerofoil profiles are demonstrated as underneath:

Four-digit series

The NACA four-digit wing sections define the profile by:

- First digit provides maximum camber which is in percentage of the total chord length.
- Second digit provides the distance of maximum camber from leading edge in tens of percentage of the chord.
- Last two digits describe maximum thickness of the airfoil as percentage of the chord.

For instance, the NACA 2412 aerofoil has a greatest camber of 2%, its position from leading edge is 40% of the chord and its maximum thickness is 12% with respect to the chord.

The NACA 0015 aerofoil is symmetrical aerofoil0, the 00 indicating that it has no camber. The 15 indicates that the airfoil has a 15% thickness to chord length ratio.

Five-digit series

The NACA five-digit series describes more complex airfoil shapes:

The first digit has to be multiplied by 0.15, providing ideal lift coefficient. The second digit has to be multiplied by 5, providing the position of greatest camber along the chord. The third digit indicates whether the camber is straightforward (0) or reflex (1).The fourth and fifth digits gives the greatest thickness of the aerofoil in percentage of the chord.

For example, the NACA 24116 profile- an airfoil with lift coefficient of 0.3, the location of of maximum camber at 20% chord, the value of reflex camber (1), and maximum thickness of 16% of chord length.

The camber-line is defined in two sections:

$$y_c = \begin{cases} \frac{k_1}{6}\left\{x^3 - 3mx^2 + m^2(3-m)x\right\}, & 0 < x < p \\ \frac{k_1 m^3}{6}(1-x), & p < x < 1 \end{cases} \tag{1.2}$$

where the chordwise location x and the ordinate y have been normalized by the chord. The constant m has the value such that maximum camber happens at x=p.Finally, constant k_1 is determined to provide the desired lift coefficient. For a camber-line profile (the first 3 numbers in the 5 digit series) $k_1 = 15.957$ is used.

1.3 TYPES OF FLOW ON AEROFOILS

Laminar stream is portrayed by layers, or laminas, of air moving at the same rate and in the same course. No fluid is traded between the laminas and the stream require not be in a straight line. The closer the laminas are to the airfoil surface the slower they move. For a perfect liquid the stream takes after the bended surface easily, in laminas. In turbulent stream, the streamlines or stream examples are muddled and there is a exchange of liquid between these ranges. Momentum is likewise traded such that slow moving liquid particles accelerate and quick moving particles surrender their energy to the slower moving particles and ease off themselves. All or about all liquid stream shows some level of turbulence.

The Reynolds number is an essential worth for the conduct of the stream, and particularly the limit layer. Streams with the same Reynolds number carry on comparative. This number can be computed by the accurate formula. If medium accuracy is sufficient, the Reynolds number can be approximately calculated by the equation given below:

$Re = v \times l \times 70000$

where: "v" is flight speed

"l" is chord length in m.

70000 constant value for air (s/m2).

The Reynolds number is in terms of a length, which is typically the chord length of an airfoil (in two measurements) or the chord length of a wing. Since the chord length of a wing may fluctuate from root to tip, a mean aerodynamic chord length is utilized to characterize the Reynolds number for a wing.

Different sorts of stream incorporate subsonic (Mach No $M < 0.1$), transonic and supersonic streams over the aerofoils. Correspondingly the fluid may b considered as compressible or incompressible or inviscid.

1.4 LITERATURE REVIEW

This section explains some previous literature on aerofoil sections and their analysis procedure.

Rana et al. [1] studied the flutter characteristics of an airfoil in a 2-D subsonic flow by using RANS based CFD solver with a structural code in time domain.

Gultop [2] contemplated the effect of viewpoint degree on Airfoil execution. The explanation behind this study was to center the swell conditions not to be kept up all through wind passage tests.

Goel[3] formulated a technique for advancement of Turbine Airfoil utilizing Quansi – 3D examination codes. He understood the intricacy of 3D demonstrating by displaying numerous 2D air foil segments and joining their figure in spiral bearing utilizing second and first order polynomials that prompts no harshness in the radial direction.

Arvind [4] looked into on NACA 4412 aerofoil and investigated its profile for thought of a plane wing .The NACA 4412 aerofoil was made utilizing CATIA V5 And examination was completed utilizing business code ANSYS 13.0 FLUENT at a rate of 340.29 m/sec for angle of attacks of 0˚, 6, 12 and 16˚. k-ε turbulence model was accepted for Airflow. Changes of static weight and element weight are plotted in type of filled shape.

Fazil and Jayakumar [5] presumed that in spite of the way that it is less requesting to model and make an aerofoil profile in CAD environment using camber billow of concentrates, after the making of vane profile it is extraordinarily troublesome to change the condition of profile for dismemberment or change reason by using surge of core inter**.**

Kevadiya [6] concentrated on the NACA 4412 aerofoil profile and remembered its significance for examination of wind turbine edge. Geometry of the aerofoil is made using GAMBIT 2.4.6. Additionally CFD examination is done using FLUENT 6.3.26 at distinctive methodologies from 0˚ to 12˚.

Guilmineau et al. [7] talked about the processing of the time-mean, turbulent, two-dimensional incompressible thick stream past an airfoil at settled rate. Another physically reliable technique is exhibited for the reproduction of speed fluxes which emerge from discrete mathematical statements for the mass and energy equalization. This conclusion strategy for fluxes makes conceivable the utilization of a cell-focused network in which speed and pressure questions have the same location, while going around the event of spurious pressure modes.

Kunz and Kroo [8] advanced aerofoils at ultra-low Reynolds numbers. These examinations are done to comprehend the aerodynamic issues identified with the low speed and miniaturized scale air vehicle outline and execution. The enhancement strategy utilized is in view of concurrent pseudo-time venturing in which stationary states are acquired by fathoming the preconditioned pseudo-stationary arrangement of comparisons.

N. Ahmed et al. [9] contemplated the numerical reproduction of stream past aerofoils is vital in the flight optimized outline of air ship wings and turbo-hardware parts. These lifting gadgets regularly achieve ideal execution at the state of onset of partition. Hence, division phenomena must be incorporated if the examination is gone for pragmatic applications. Thus, in the present study, numerical recreation of relentless stream in a straight course of NACA 0012 aerofoils is expert with control volume approach.

Mittal et al. [10] performed the computational examination for two-dimensional stream past stationary NACA 0012 aerofoil is completed with dynamically expanding and diminishing approaches. The incompressible, Reynolds averaged Navier–Stokes mathematical statements in conjunction with the Baldwin–Lomax model, for turbulence conclusion, are explained utilizing settled limited element formulations.

Several experimental studies were also conducted to understand the dynamic behaviour and flow characteristics of aerofoil sections.

Genc et al [11] conducted experiments on NACA 2415 aerofoil by varying angle of attack from -12^0 to 20^0 at low Reynolds number flight regime ($0.5x10^5$ to $3x10^5$). Using pitot static tube, scan valve and pressure transducer, the pressure distribution over aerofoil was measured. Lift, drag and pitching moment were obtained by 3-component load cell system. Hot wire anemometer and oil flow visualization was used to photograph surface flow patterns.

1.5 OBJECTIVE AND SCOPE OF PRESENT WORK

The objective of this project is to understand the phenomena of the uniqueness of aerofoil shape. Aerofoil shapes are employed in aircraft sectors as well as in automobile and production sectors e.g. wind turbines, wing of an automobile etc. It can generate lift as well as down force when used in a specific manner. So it is quite important to decode the phenomena behind its shape and the process by which it produces necessary lift and down force.

The main objective is to study the design process of various aerofoils and their flow simulation to understand how they work. Due to time limitation, the following are the essential objectives of the work:

- Obtain the pressure distribution, lift and drag coefficients on the given aerofoil profile using CFD solver fluent at various angles of attack in in-viscid domain.
- Use the panel method for comparison of the numerical results obtained from FLUENT.
- Future scope of employing the inverse method to get the desired aerofoil shape according to given parameters.

CHAPTER 2
MATHEMATICAL ANALYSIS

This chapter explains the method of generation of aerofoils, fluid analysis and Panel method used for 2-D aerofoils.

2.1 AEROFOIL PROFILE GENERATION

Aerofoil cross-sections are asymmetric in nature, where the shear centre and centre of flexure are not coincident. Often it requires an integration of geometry modeller, mesh generator and CFD solver. The asymmetric models of NACA airfoil series is defined by its digits e.g. NACA 2412, which describe thickness, maximum amount of camber and its position from leading edge. For example if an airfoil number is

NACA MPXX

e.g.

NACA 2314

then:

M is the greatest camber which has to be divided by 100. M=2 therefore the camber is 0.02 or 2% chord length

P is called the position of the greatest camber which is divided by 10. P=3 therefore it is concluded the maximum camber is at 0.3 or 30% of the chord.

XX is the thickness divided by 100. XX=14 so the thickness is 0.14 or 14% of the chord.

NACA symmetric aerofoils are denoted by NACA 00XX where the last two digits provide the ratio between greatest thickness of aerofoil with respect to chordlength.

The NACA airfoil cross-section is derived from a camber line and a thickness distribution plotted perpendicular to the camber line.

The comparison for the camber line is part into either side of the purpose of most extreme camber position. To compute the position of the last aerofoil envelope the inclination of the camber line is likewise needed.

The relations for camber line are:

$y_c = Z_c/c = \quad m/p^2(2px-x^2) \quad$ for $0 \leq x \leq p$

$m/(1-p)^2(1-2p+2px-x^2 \quad$ for $p \leq x \leq 1$ (2.1)

The thickness distribution is given by the equation:

$$y_t = \frac{T}{0.2}\left(a_0 x^{0.5} + a_1 x + a_2 x^2 + a_3 x^3 + a_4 x^4\right)$$

Where:

$a_0 = 0.2969 \quad a_1 = -0.126 \quad a_2 = -0.3516 \quad a_3 = 0.2843$

$a_4 = -0.1015$ or -0.1036 for a closed trailing edge

(2.2)

Here, y_t is distance from centre line, t is maximum thickness from centre line, c is the chord length, x= X/c is positive along the chord and varies from 0 to 1

The constants a_0 to a_4 are for a 20% thick airfoil. The expression t/0.2 adjusts the constants to the required thickness.

At the trailing edge (x=1) there is a thickness of 0.0021 chord width for a 20% aerofoil. In the event that a shut trailing edge is obliged the estimation of a4 is changed in accordance with 0.1036 as it gives the littlest bother of the surface bend.

The estimation of y_t is a half thickness and is connected both sides of the camber line.

Utilizing the comparisons, for a given estimation of x it is conceivable to figure the camber line position, the inclination of the camber line and the thickness. The position of the upper and lower surface can then be calculated perpendicular to the camber line, according to:

Upper surface $x_u = x_c - y_t \sin(\theta)$ $y_u = y_c - y_t \cos(\theta)$

Lower surface $x_l = x_c + y_t \sin(\theta)$ $y_l = y_c - y_t \cos(\theta)$ (2.3)

Where $\tan\theta = (2m/p)\,(p-x)$, $0 \le x \le p$

$= (2m/(1-p)^2)(p-x)$, $p \le x \le 1$ (2.4)

The most clear approach to plot the airfoil is to repeat through similarly dispersed estimations of x figuring the upper and lower surface directions. The focuses are all the more generally dispersed around the main edge where the curve is most noteworthy. To gathering the focuses at the finishes of the airfoil segments cosine dividing is utilized with uniform augmentations of ß

$$x = \frac{(1-\cos(\beta))}{2} \quad \text{where: } 0 \le \beta \le \pi \tag{2.5}$$

A popular website called NACA Airfoil profile generator was used which has these in-built mathematical functions programmed in it. Here we just have to provide the necessary data e.g. camber, thickness etc. and the software generates the profile and produces the coordinates of the profile. Fig 2 shows the profile obtained for NACA 2314 series from a simple code based on above equations in MATLAB.

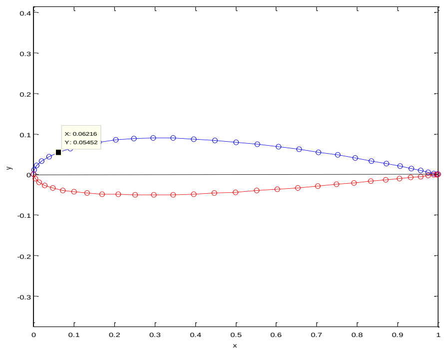

Fig 2 NACA 2314 series profile

2.2 NUMERICAL ANALYSIS OF FLOW BEHAVIOUR (CFD)

Flow over aerofoil is mathematically described by Navier-Stokes equation as follows:

$\frac{\partial u}{\partial t}$ + ρ (u.∇)u - ∇.σ(u.p)=ρf

and ∇.u=0,

with ρ as fluid density, f being the body force.

CFD is a numerical technique used to recreate physical issues with utilization of fluid comparisons. This methodology is utilized to research plan without making a physical model – and can be an important to comprehend properties of new mechanical plans. By utilizing a simulation as opposed to doing lab tries, one may get results faster. A vital thing in the utilization of CFD is to comprehend the improvements in programming, and know the constraints. In spite of the fact that the CFD programming uses surely understood representing comparisons, serious disentanglements are made as far as matrix and representing geometries.

Following is the method employed to carry through the CFD simulation:

- Prepare geometric model.
- Meshing
- Boundaries
- Software (Fluent) set up, initialization and solving.

Preparing the model

NACA airfoils geometries were acquired as co-ordinate vertices i.e. writings document and imported into the ANSYS programming. Some minor conformities were made to this to right the geometry and make it substantial as a CFD model.

ANSYS is fundamental during the time spent doing the CFD investigation: it creates the workplace where the item is reproduced. An imperative part in this is making the cross section encompassing the article. This needs to be reached out in all bearings to get the physical properties of the encompassing fluid – for this case moving air. The mesh and edges should likewise be gathered keeping in mind the end goal to define the essential limit conditions adequately.

Meshing

A situation comprising of 2 squares and 1 half circle encompasses the NACA airfoil. The cross section is developed to be fine at areas near to the airfoil and with high vitality, and coarser more remote far from the airfoil. For this airfoil an organized quadratic lattice was utilized.

The lattice must be fine likewise in specific districts a long way from the airfoil. A fine work infers a higher number of counts which thus makes the simulation long. For the NACA airfoils, the exceptionally front has an edge network conveyed with an expanding separation between hubs, beginning from little sizes.

Setting Boundary conditions

Giving properties to the distinctive geometries is essential for a recreation. For this situation, the cross section limits were offered situated to the x and y velocity components, and the end limit the property "pressure outlet" to reproduce the zero gauge pressure.

Setting up Fluent

The geometries and mesh were imported into FLUENT, and the framework and environment properties were situated. "Double precision" is chosen as framework parameters, guaranteeing sufficient exactness. FLUENT has single exactness as default, however for these reproductions a precise arrangement is asked. The residuals for the distinctive turbulence model variables were situated to 10-8 and the cycle max check to 1000. The simulation procedure could likewise be stopped or ceased if the CL or CD appeared to have balanced out appropriately.

Viscosity models

Contemplating the vortex property and limit layer partition for airfoils, this simulation will need to manage turbulent streams. The disorderly way of turbulent stream makes it extremely hard to figure speeds for all focuses in space. RANS is the inverses to DNS which is the explanatory direct simulation of the overseeing mathematical statements, and utilize a measurable and arrived at the midpoint of way to deal with discover the stream conduct. The purpose behind utilizing RANS models is that little vortices are uprooted by averaging the stream.

A pivotal point is selecting a viscous model, and in FLUENT there are a few choices. There are basic contrasts to the distinctive models, and may be utilized for diverse sorts of streams. The screenshot of the fluent environment for general case is depicted in Fig.3

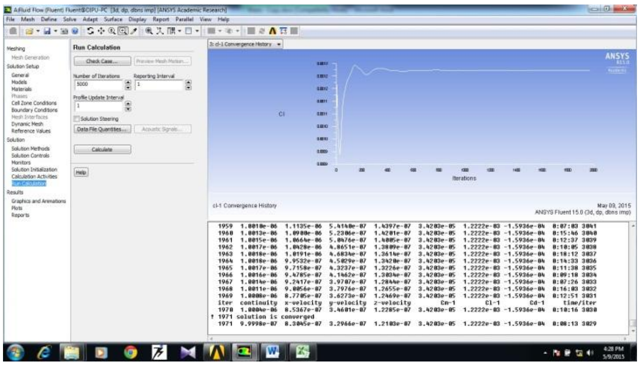

Fig.3 Fluent environment

2.3 PANEL METHOD

Panel methods are popular ways for solving incompressible fluid flow over a 2-D and 3-D geometries.

In 2-D, the airfoil surface is generally divided into straight line segments or panels. On each panel are placed vortex sheets of strength g.

Vortex sheets are used because (miniature vortices strength= $\gamma_0 \partial s$, ∂s being length of a panel) vortices give rise to circulation, which in turn produce lift.

Panel method considers the airfoil as a series of straight line segments as shown in the Fig. 4

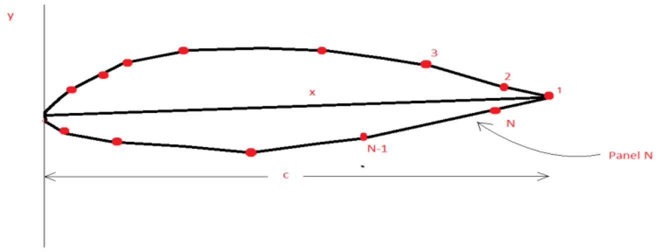

Fig 4 the aerofoil as a no of panels

Every panel has two end focuses (board joints) and is characterized by the control point, at the board focus, where we apply the vital limit condition y=C.

At the point when there are more the quantity of panels, then the more exact is the arrangement, since a continuous curve is represented to by arrangement of straight lines.

The aerofoil is constantly considered as stream line surface

This verifies that the speed of aerofoil stays tangential to the surface of aerofoil.

At all control focuses (centre purposes of every panel) y= C

The stream capacity is characterized as the superposition of the impacts of the free stream and the impacts of vortices.

At first the aerofoil is isolated into N boards. A panel is given the number j, where J differs from 1 to N.

We expect that γ_0 is a steady on every board. Hence, on a board j, the obscure quality is γ_0j

The control points arranged at the focuses of every board are numbered. Every control point is given the symbol "i", where i changes from 1 to N.

The integral equation becomes

$$u_\infty y_i - v_\infty x_i - \sum_{j=1..N} \frac{\gamma_{0,j}}{2\pi} \int_j \ln\left(\left|\bar{r}_i - \bar{r}_0\right|\right) ds_o - C = 0$$

We make use of two indices ‘i’ and ‘j’. The index ‘i’ points to the control point where equation is applied.

- The index ‘j’ points to the panel over which the line integral is evaluated.
- The integrals computed over the individual panels are only dependent on the panel shape (straight line segment), its end points and the control point í’.
- We refer to the resulting quantity as

$$u_\infty y_i - v_\infty x_i - \sum_{j=1}^{N} A_{i,j} \gamma_{0j} - C = 0$$

where,

$$A_{i,j} = \text{Influence of Panel j on index i} = \frac{1}{2\pi} \int \ln\left(\left|\bar{r}_i - \bar{r}_0\right|\right) ds_0$$

- N+1 equations are obtained for the unknowns $g_{0,j}$ (j=1…N) and C.

It was assumed that the first panel (j=1) and last panel (j=N) were on the lower and upper surface trailing edges

$$u_\infty y_i - v_\infty x_i - \sum_{j=1}^{N} A_{i,j} \gamma_{0j} - C = 0$$

$$\gamma_{0,1} = -\gamma_{0,N}$$

These linear set of equations are then solved easily, and γ_0 was found out from where pressure and pressure coefficient C_p were found out. Once distribution strengths were calculated, the surface tangential velocities at the centre of the each panel Vj is computed and the the surface pressure corefficients Cpj= (1- $V i2 / V\alpha 2$). The lift coefficient can be calculated by assuming small angle of attack according to the following equation:\

$C_L = \sum^{N}_{i=1} Cp_i(x_i-x_{i+1})/c$

The pitching moment coefficient can be also similarly calculated as the sum of the panel moments about the one-fourth chord-

$Cm= \sum i=1^{N} Cp_i(x_i-x_{i+1})/c \ \ (((x_{i+1}-x_i)/2c)-(1/4))$.

CHAPTER 3

RESULTS AND DISCUSSION

3.1 Obtaining the coordinates of NACA 2314 aerofoil.

These coordinates have been obtained through the use of the equations which govern the NACA 4 series aerofoils. First the mean camber line was determined. After that both the upper and lower line coordinates were found out using the necessary equations and the profile was determined. Using the equations of the profile, a MATLAB code is generated which takes interactively the no of points on the profile (both top and bottom surfaces). Also user has to enter the series number and the program plots the aerofoil and stores the points in excel sheet for importing into ANSYS. Following code is the main part of the program.

```
iaf.desig='2314';

% desig='0012';
iaf.n=30;
iaf.halfcos=1;
iaf.needfile=1;
iaf.datfile='./'; this folder
iaf.is_finiteTE=0;
af = naca4seriesgen(iaf);
% plot(rf.x,rf.y,'to-')
plot(rf.xu,rf.yu,'to-')
hold on
plot(rf.xl,rf.yl,'po-')
axis equal
```

Fig 5 shows the NACA 2314 profile generated with 2 different no of points.

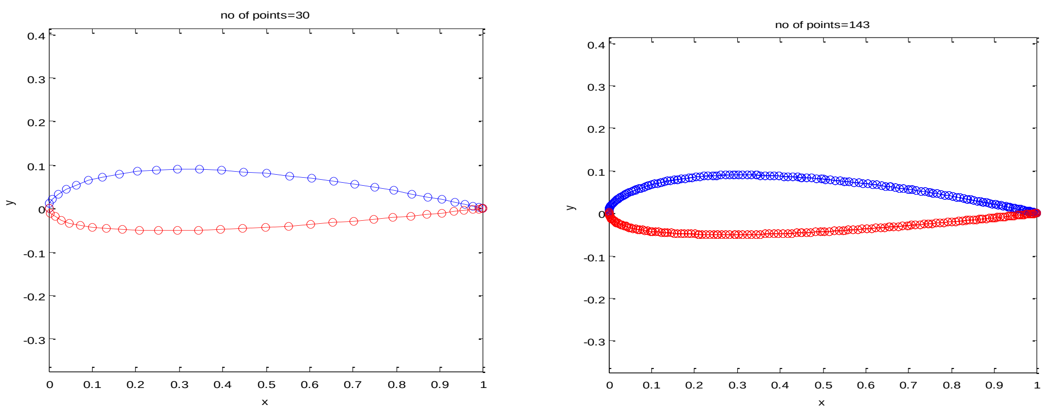

Fig. 5 NACA 2314 profiles with two diff no of control points

3. 2 Fluent results

3.2.1 Mesh Structure

Fig. 6 shows the mesh of the aerofoil with C domain. The mapped meshing is created on entire domain. The cross section is developed to be fine at areas near to the aerofoil and coarser more remote far from the aerofoil. For this aerofoil a quadratic element was utilized. The mesh has to be fine also in certain regions far from the aerofoil.

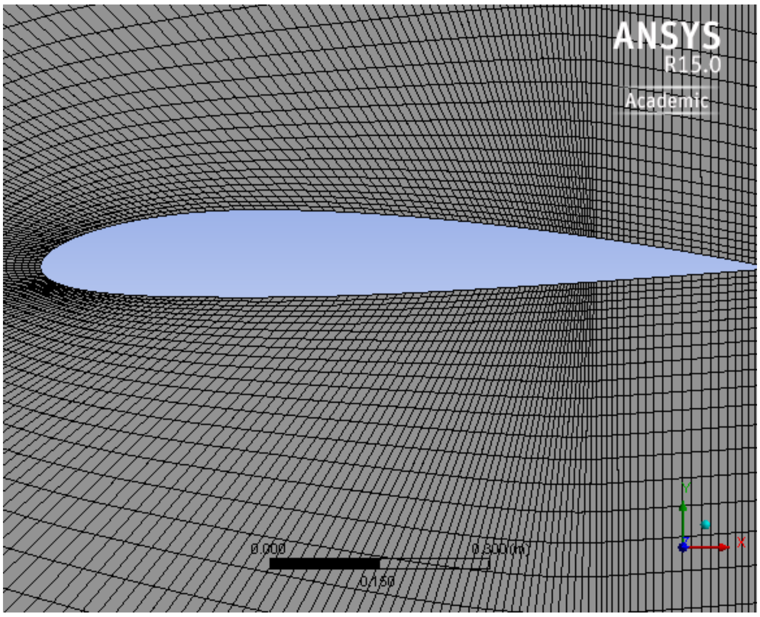

Fig.6: Mesh structure of NACA 2314

3.2.2 Pressure and velocity distribution

The NACA 2314 airfoil is tested under inviscid flow condition against a various angle of attacks such as 4, 8, and 12 degree. The pressure contour and velocity vectors are plotted and shown in figures (Fig. 7-12). The velocity vector gives the clear picture of the velocity distribution and the pressure contour gives clear picture of static pressure over an airfoil. The pressure coefficient over an airfoil is also plotted with help of Fluent and it is compared with panel method result and shown in Fig. 16-18.

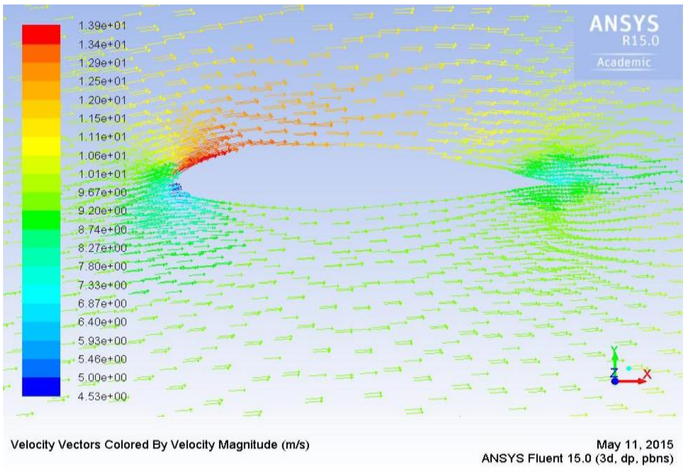

Fig. 7: velocity contours at 4 degrees angle of attack

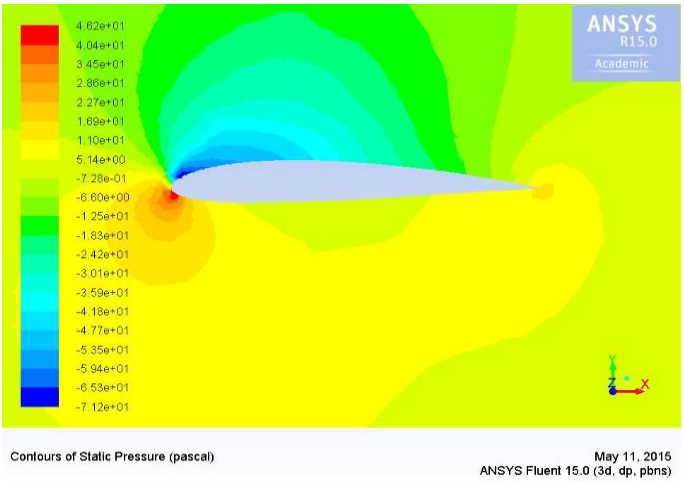

Fig. 8: pressure contours at 4 degree angle of attack

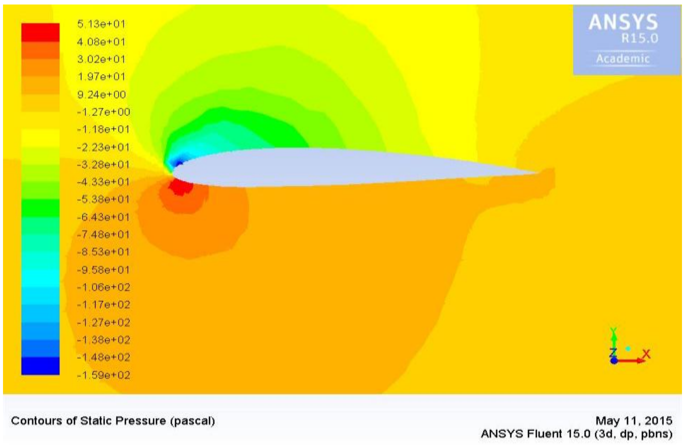

Fig. 9: pressure contours at 8 degree angle of attack

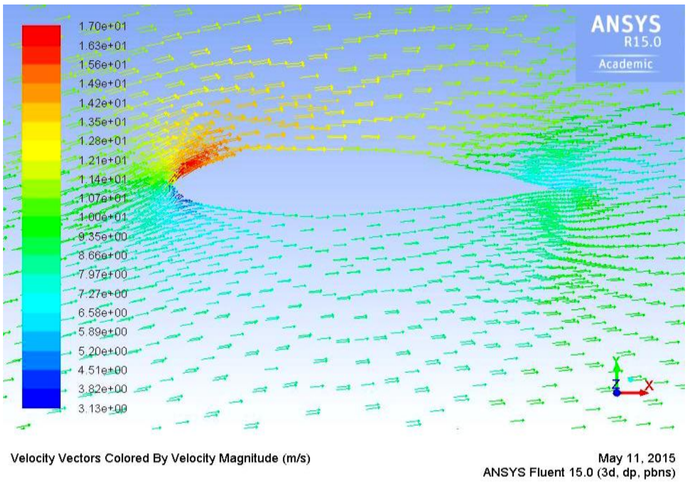

Fig. 10: velocity contours at 8 degrees angle of attack

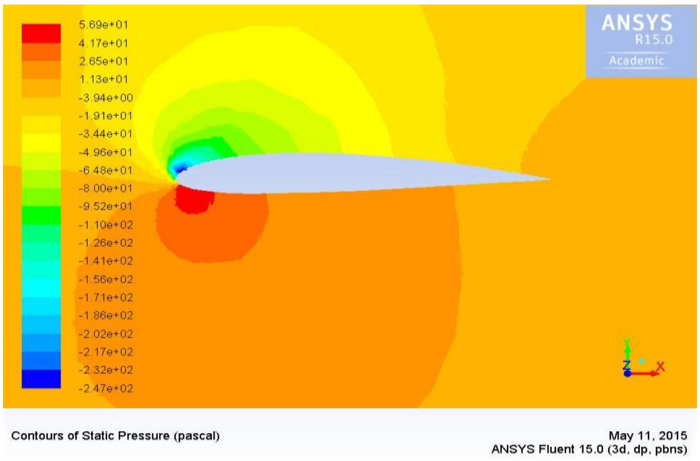

Fig. 11: pressure contours at 12 degree angle of attack

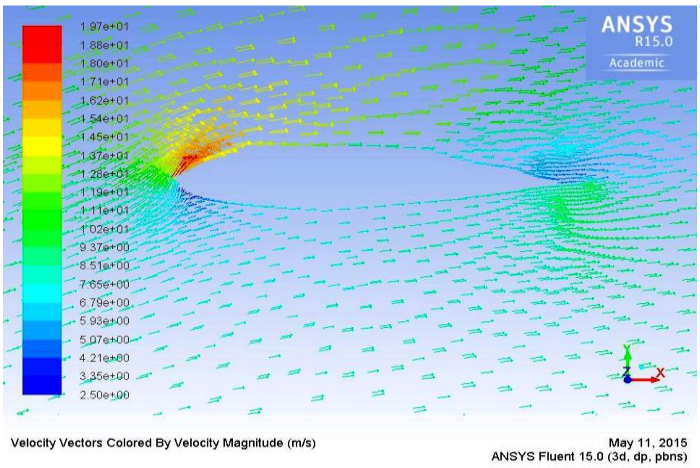

Fig. 12: velocity vectors at 12 degrees angle of attack

Table 1: the values of CL CD and CL/Cd ratios for various angles of attacks

S.No	alpha	CL	CD	Cl/Cd
1	0.0	0.21360	0.00600	35.60000
2	2.0	0.45890	0.00700	65.55714
3	4.0	0.70550	0.00807	87.42255
4	6.0	0.90060	0.00933	96.52733
5	8.0	1.08650	0.01244	87.33923
6	10.0	1.25490	0.01687	74.38648
7	12.0	1.37130	0.02280	60.14474
8	14.0	1.42900	0.03460	41.30058
9	16.0	1.40020	0.06199	22.58751
10	18.0	1.34640	0.09803	13.73457
11	20.0	1.27620	0.13937	9.15692

The lift to drag ratio is an important parameter in measuring the airfoil performance. When L/D ratio is high it shows the drag coefficient has very less value. The L/D ratio for NACA 2314 is shown in Fig. 13.

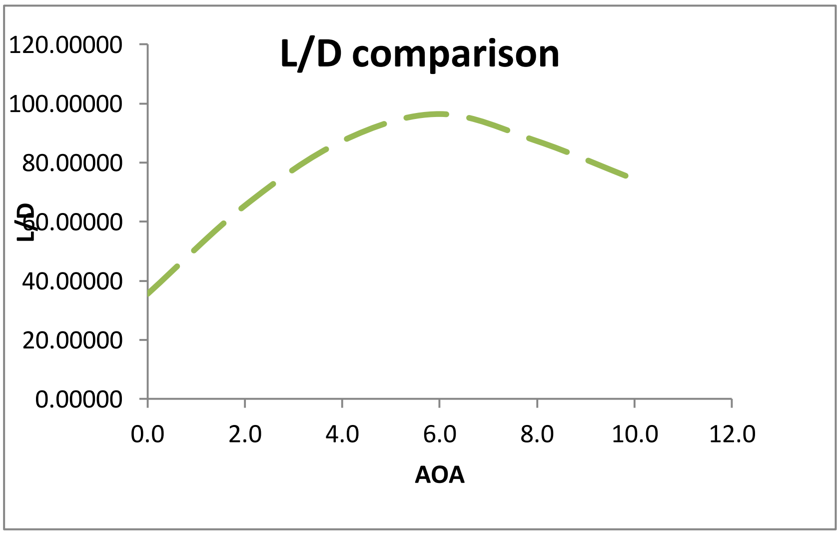

Fig.13: L/D comparison for different angle of attacks

3.3 Panel Method Results

The Fig. 14 shows the aerofoil with 34 panels and ready to be used for mathematical analysis. This was a result of a MATLAB code written considering all the equations and conditions of panel method. The output of panel method such as coefficient of pressure various angle of attacks such as 4, 8 and 12 degrees and coefficient of lift for range of AOA were plotted and shown in Fig. 15-18.

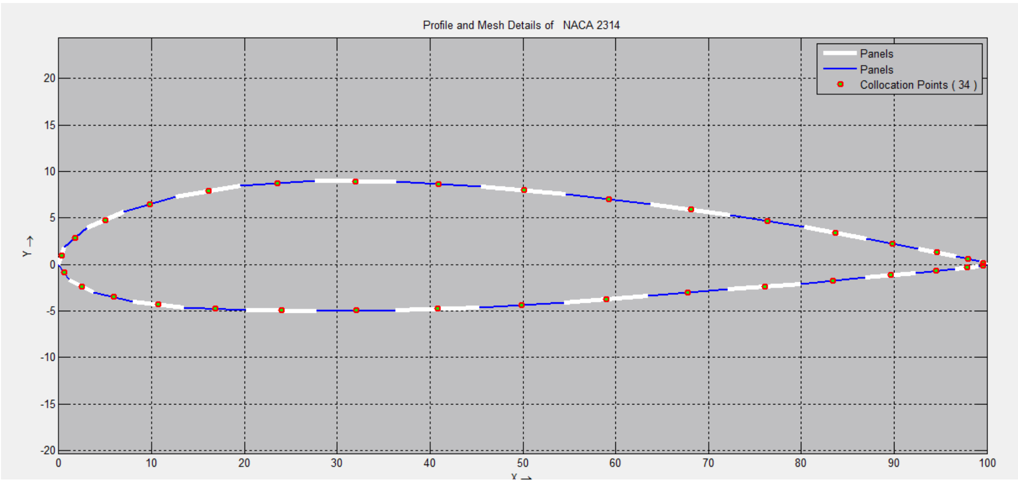

Fig.14: aerofoil divided into panels.

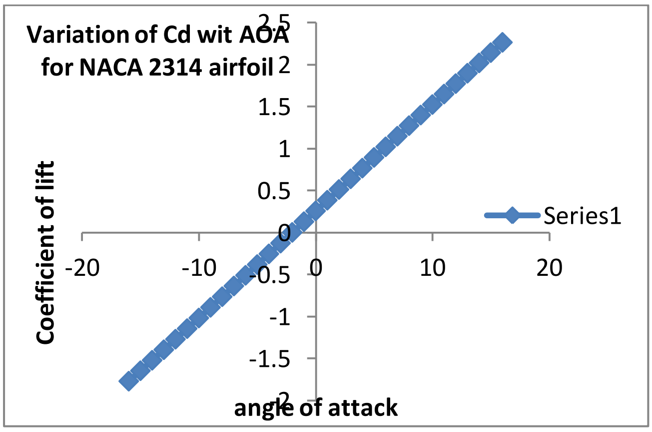

Fig.15: Variation of Cd with angle of attack.

As we have considered an inviscid model, so that Cl will keep on increasing without any signs of stall as we increase angle of attack.

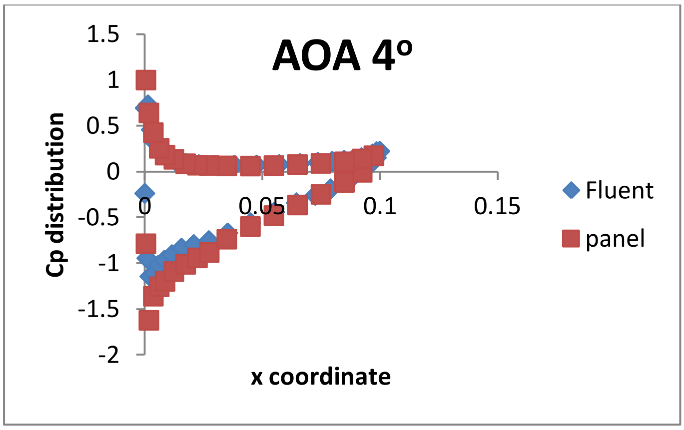

Fig. 16: C_p distribution at 4 degree angle of attack

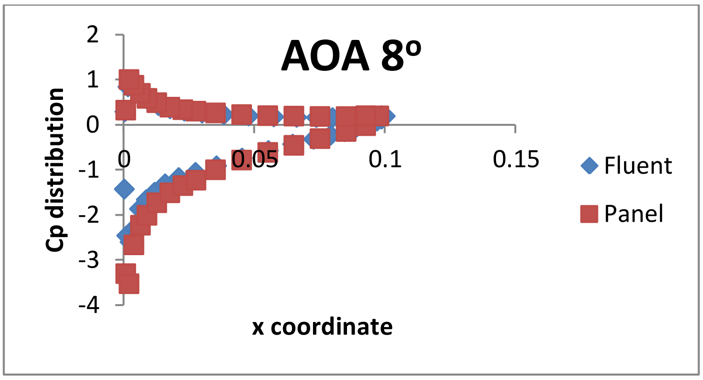

Fig. 17: C_p distribution at 8 degree angle of attack

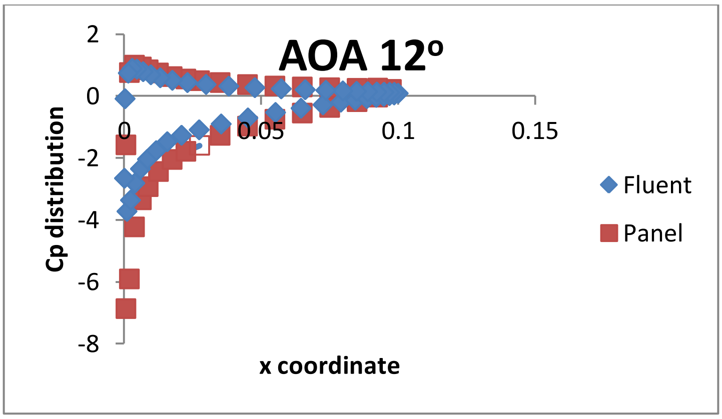

Fig. 18: Cp distribution at 12 degree angle of attack

The comparison of c_p distribution by panel method and Fluent is plotted for various AOA and it is observed that error between two methods is very less. So the panel method can be employed in inverse design of airfoils through optimization in future work.

CHAPTER 4
CONCLUSION

4.1 SUMMARY

In this work, NACA 2314 aerofoil cross-section was studied and the principles governing the shape and size were thoroughly investigated. Also the coefficient of lift, drag and pressure were determined for an aerofoil. The performance of an airfoil was studied thoroughly by panel method (inviscid); first the coordinates of an aerofoil were determined using the NACA four series equations and the airfoil was divided into number of panels and flow characteristics over a panel and overall lift coefficient were achieved. Then ANSYS Fluent analysis were carried on the same model at different angle of attacks and the velocity and pressure distribution as well as the lift and drag force were determined and compared. Panel method optimization was used in this to optimize the provided parameters according to required parameters. A MATLAB code was used to achieve this result. And at last coordinates of the optimized shape of the aerofoil was obtained.

4.2 FUTURE SCOPE

As the future scope the present model can be investigated in viscous and compressible region to study its behaviour thoroughly and the designing of an airfoil according to its specific applications will be carried out with different optimizations scheme along with inverse method.

REFERENCES

[1]. D.Rana, S.Patel, AK.Onkar and M.Manjuprasad, “Time domain simulation of airfoil flutter using fluid structure coupling through FEM based CFD solver”, Symposium of Apllied Aerodynamics and Design of Aerospace vehicle, SAROD20111, Nov 16-18 2009, Bangalore

[2]. T. Gultop , “An Investigation of the effect of aspect ratio on Airfoil performance”, Gazi : American Journal of Applied Sciences, Vol: 2, Issue: 2: 545-549 ,1995

[3]. S.Goel “Turbine Airfoil Optimization Using Quasi-3D Analysis Codes.” International Journal of Aerospace Engineering ISSN/EISSN: 16875974 16875974 Volume: 2, 2008

[4]. M. Arvind **“**CFD ANALYSIS OF STATIC PRESSURE AND DYNAMIC PRESSURE FOR NACA 4412” International Journal of Engineering Trends and Technology ISSN/EISSN: 22315381 Vol: 4 Issue: 8 Pages: 3258-3265,(2010)

[5]. J. Fazil and V. Jayakumar, “Investigation of airfoil profile design using reverse engineering Bezier curve” Journal of Engineering and Applied SciencesISSN/EISSN: 18196608 Year: 2011 Volume: 6 Issue: 7 Pages: 43-52, 2011.

[6]. Kevadiya M. (2013) CFD “Analysis of Pressure Coefficient for NACA 4412” International Journal of Engineering Trends and Technology ISSN/EISSN: 22315381 Volume: 4 Issue: 5 Pages: 2041-2043, 2013

[7]. E.Guilmineau, J. Piquet & P. Queutey “ 2D turbulent viscous flow simulation past airfoils at fixed incidence” Comp. Fluids, Vol.26, pp.135-162., 1997

[8]. Kunz, P. J., Kroo, I., “Analysis, design and testing of airfoils for use at ultra-low Reynolds numbers, Proceedings of the Conference on Fised, Flapping and Rotary Vehicles at very Low Reynolds Numbers”, edited by T. J. Mueller, Univ. of Notre Dame, Notre Dame, IN, pp.349-372, 2000.

[9]. N. Ahmed, B.S. Yilbas*, M.O. Budair; “Computational study into the flow field developed around a cascade of NACA 0012 airfoils.”” Comput. Methods Appl. Mech. Engrg. 167 17-32. 1998.

[10]. S. Mittal and P. Saxena; “Hysteresis in flow past a NACA 0012 airfoil”. Comput. methods Appl. Mech. Engrg. 191 2179– 2189, 2002

[11]. M. S. Genc, I. Karasu, H. H. Acikel, “An experimental study on aerodynamics of NACA 2415 aerofoil art low Reynolds numbers”, Exp. Thermal and fluid science, vol.39, pp. 252-264, 2012

Web references

- http://www.airfoiltools.com (9 th may, 2015)
- http://www.boeing.com/boeing/commercial/737family/ (5th May, 2015)
- http://www.dcs-mig21.com/ (5th may, 2015)
- http://www.aviastar.org/helicopters_eng/bell_xv-3.php (3rd May, 2015)
- http://aerospace.illinois.edu/m-selig/ads/coord_database.html (3rd May, 2015)
- http://www.aerospaceweb.org/question/airfoils/q0041.shtml (3rd May, 2015)

APPENDIX

Code for calculating the coordinates of the desired aerofoil section in MATLAB

```
t=str2num(iaf.designation(3:4))/100;
m=str2num(iaf.designation(1))/100;
p=str2num(iaf.designation(2))/10;

a0= 0.2969;
a1=-0.1260;
a2=-0.3516;
a3= 0.2843;

if iaf.is_finiteTE ==1
    a4=-0.1015; % For finite thick TE
else
    a4=-0.1036;  % For zero thick TE
end

% % [[Giving x-spacing---------------------------------------------------]]
if iaf.HalfCosineSpacing==1
    beta=linspace(0,pi,iaf.n+1)';
    x=(0.5*(1-cos(beta))); % Half cosine based spacing
    iaf.header=['NACA' iaf.designation ' : [' num2str(2*iaf.n) 'panels,Half
cosine x-spacing]'];
else
    x=linspace(0,1,iaf.n+1)';
    iaf.header=['NACA' iaf.designation ' : [' num2str(2*iaf.n)
'panels,Uniform x-spacing]'];
end

yt=(t/0.2)*(a0*sqrt(x)+a1*x+a2*x.^2+a3*x.^3+a4*x.^4);

xc1=x(find(x<=p));
xc2=x(find(x>p));
xc=[xc1 ; xc2];

if p==0
    xu=x;
    yu=yt;

    xl=x;
    yl=-yt;

    zc=zeros(size(xc));
else
    yc1=(m/p^2)*(2*p*xc1-xc1.^2);
    yc2=(m/(1-p)^2)*((1-2*p)+2*p*xc2-xc2.^2);
    zc=[yc1 ; yc2];

    dyc1_dx=(m/p^2)*(2*p-2*xc1);
    dyc2_dx=(m/(1-p)^2)*(2*p-2*xc2);
    dyc_dx=[dyc1_dx ; dyc2_dx];
```

```
    theta=atan(dyc_dx);

    xu=x-yt.*sin(theta);
    yu=zc+yt.*cos(theta);

    xl=x+yt.*sin(theta);
    yl=zc-yt.*cos(theta);
end
af.name=['NACA ' iaf.designation];

af.x=[flipud(xu) ; xl(2:end)];
af.z=[flipud(yu) ; yl(2:end)];

indx1=1:min( find(af.x==min(af.x)) );  % Upper surface indices
indx2=min( find(af.x==min(af.x)) ):length(af.x); % Lower surface indices
af.xU=af.x(indx1); % Upper Surface x
af.zU=af.z(indx1); % Upper Surface z
af.xL=af.x(indx2); % Lower Surface x
af.zL=af.z(indx2); % Lower Surface z

af.xC=xc;
af.zC=zc;

lecirFactor=0.8;
af.rLE=0.5*(a0*t/0.2)^2;

le_offs=0.5/100;
dyc_dx_le=(m/p^2)*( 2*p-2*le_offs );
theta_le=atan(dyc_dx_le);
af.xLEcenter=af.rLE*cos(theta_le);
af.yLEcenter=af.rLE*sin(theta_le);

% % [[Writing iaf data into file-------------------------------------------
]]
if iaf.wantFile==1
    F1=iaf.header;
    F2=num2str([af.x af.z]);
    F=strvcat(F1,F2);
    fileName=[iaf.datFilePath 'naca' iaf.designation '.dat'];
    dlmwrite(fileName,F,'delimiter','')
end
```